NUMERICAL INVESTIGATIONS IN THREE-DIMENSIONAL INTERNAL FLOWS

SEMI-ANNUAL STATUS REPORT

1 APRIL THROUGH 30 SEPTEMBER 1988

Prepared for:

NASA-AMES RESEARCH CENTER

MOFFETT FIELD, CA   94035

UNDER NASA GRANT

NCC 2-507

by:

WILLIAM C. ROSE

ENGINEERING RESEARCH AND DEVELOPMENT CENTER

UNIVERSITY OF NEVADA, RENO

RENO, NV   89557

UNR SEMI-ANNUAL STATUS REPORT
1 APRIL THROUGH 30 SEPTEMBER 1988

I. BACKGROUND

Renewed interest in simulation of high enthalpy flows has been brought about by interest in the design and testing of hypersonic propulsion components. This renewed interest has led to refurbishment of high speed ground based wind tunnel facilities. The National Aerospace Plane (NASP) Program is one requiring the use of such ground based facilities for many aspects of vehicle testing. Simulation of the high enthalpy flows occurring in the NASP propulsion system is of particular interest. Studying the combustor presents some of the more challenging problems in the ground based simulation of the fluid flow. Impulse facilities, that is facilities whose steady state running time is less than approximately a second, have been shown recently to be able to produce credible simulations of the required aerodynamic flow environment. One such facility is available at the NASA-Ames Research Center in a reflected-shock-wave wind tunnel. The nozzle for the facility was designed for operation at near Mach 7, and relies on high stagnation pressures and temperatures achieved by the reflection of a shock wave at the end of a driven shock tube. Total pressures up to approximately 8000 psi (55 MPa) and total temperatures of up to 9000°R (5000°K) are obtainable.

A difficulty in simulating the required conditions at the entrance to the combustor is related to achieving a sufficiently high static pressure such that combustion would be possible at a high Mach number. It is presently thought that at least a half an atmosphere static pressure is required for the efficient combustion of

2

gaseous hydrogen. The conditions described above will allow the static pressure to be high enough only if the Mach number in the simulation is reduced somewhat. An analysis of the flow within the original nozzle has not been carried out previously using numerical simulations, although it has been operated experimentally. The flow field that could be expected to arise by the reduced Mach number requirement has not been examined previously either analytically or experimentally and is of considerable interest in the present study. A Navier-Stokes code has been chosen in the present study to examine analytically the flow within representative selected nozzles for use in combustion experiments.

The numerical simulation of these fluid flows is popular and may be able to shed light on the direction that redesign of the shock tube nozzle should take. The numerical simulations are sensitive to the assumptions used in setting up the problem. Because of the very high temperatures involved, certain so-called real gas effects may be important and they must be taken into consideration. In the present study, the numerical codes have been modified to account for the axially symmetric flow field, for the inclusion of equilibrium real gas effects (taking account of molecular vibration) and for ideal gases whose thermodynamic properties are frozen at the stagnation conditions. This study investigates numerical simulation of flows within the original and two proposed nozzles, including the energy lost to the relatively cold walls.

II. INTRODUCTION

In the NASA-Ames reflected-shock-wave wind tunnel, the conditions in the driven tube are variable to produce a range of stagnation pressures and stagnation temperatures. The nominal length of the nozzle and its final diameter are currently assumed to be fixed by existing hardware. Any candidate geometry for a redesigned nozzle must accommodate the range of stagnation conditions and be physically positionable within the confines of the existing hardware. Thus, the free parameter when variations in Mach number (which is related to final static pressure) are considered is the diameter of the throat. The initial simulation carried out here involves the nominal Mach 7 contours with a moderate enthalpy condition. It is known experimentally that the flow field at the exit of this nozzle is about one-third boundary layer flow and two-thirds is a core flow. The approach in the present study is to conduct a numerical simulation of this nozzle and examine the calculated variables in light of knowledge about the actual flow field within the facility. Because of the desirability to carry out a merged viscous and inviscid flow field, and furthermore to examine the potential for separations that might be present within the contraction section of the nozzle, the full two-dimensional Navier-Stokes code discussed in the previous status report was used in the present study. This code has been modified in the present work to allow for various thermodynamic assumptions within the code. In addition, the originally planar two-dimensional code has been modified to account for axially symmetric flow in radial coordinates. Because the flow is assumed to be axially symmetric, no requirement exists for using a three-dimensional code.

III. RESULTS AND DISCUSSION

Initially, the Navier-Stokes code was used to carry out a simulation of the Mach 7 reflective shock tube nozzle. A mesh was generated that had 201 points in the streamwise direction by 31 points in the radial direction. In the radial direction the mesh was compressed near the solid surface in order to resolve the viscous layer. The lower boundary of the simulation is the nozzle centerline. In addition to the compression in the radial direction, the grid was made more dense near the throat of the nozzle than towards the exit of the nozzle in order to obtain what was thought to be the required resolution of the flow within the throat. Results of the simulations are presented in this report in terms of Mach number and total temperature contours, and, at times, velocity vector plots for studying the nature of the flow within various nozzles. Figure 1 depicts the Mach number contours for the simulation for the original nozzle. Figure 1a shows the Mach number contours presented on the scale geometry, that is both the Y and X scales are the same, so that a true representation of the size of the nozzle is shown. In order to better depict the variation of Mach number throughout the nozzle, the Y scale has been expanded in Figure 1b to a value that is five times that shown in Figure 1a while the X coordinate remains unchanged. In Figure 1b, two unexpected changes in the slope of the wall occur which are the result of a minor error in the original contours provided. In spite of these minor changes in the slope of the nozzle wall, several interesting features of the flow field can be discerned. Initially, a core Mach number near 6.6 is seen at the exit of the nozzle. A viscous layer appears to extend approximately half of the distance across the nozzle. This is in nominal agreement with the flow found experimentally by NASA

personnel. The total temperature contours non-dimensionalized by the upstream stagnation total temperature are shown in Figure 1c and indicate that some total energy is lost through heat transfer at the wall. From Figure 1c it can be concluded that most of the energy is lost near the throat of the nozzle, as would be expected on the basis of experimental data and calculations carried out earlier under this same grant (Semi-Annual Status Report 1 October 1987 through 31 March 1988). These calculations have been carried out under the assumption that the stagnation temperature was $6100^\circ R$ and that the total pressure corresponded to approximately 1500 psi. The wall temperature was assumed to remain constant at a fixed value of $1000^\circ R$. This solution was obtained under the assumption that the gas remains an ideal gas whose gamma was frozen at the stagnation value of 1.244. The time-dependent code was run until a nominally steady state solution was obtained for the entire 201 by 31 mesh.

After the solution depicted in Figure 1 was completed, an equilibrium real gas code that made simple account of the vibrational energy available to diatomic molecules was finished. Also at this time, discussions with NASA personnel indicated a higher level of interest in a hypothetical nozzle with a much larger throat (to produce the required half atmosphere static pressure condition in the test section). Accompanying the higher static pressure, the shock reflection conditions were assumed to be capable of providing increased temperature and total pressure to values of $9460^\circ R$ and 8125 psi, respectively. It was also judged that the wall temperature should be lowered from $1000^\circ R$ to $600^\circ R$ and used throughout the remainder of the simulation studies. In addition, a procedure was implemented to shift out of the active

solution the previously converged solution in the upstream portion of the nozzle, thus substantially reducing the overall computation time. This shifting procedure is equivalent to block parabolization and was used for the remaining solutions shown here.

Figure 2 shows the application of the equilibrium real gas code to the high enthalpy, very cold wall conditions for a new nozzle whose outflow and inflow radii are the same as those investigated previously, but with a much larger throat radius. This throat radius is expected to reduce the nozzle's exit Mach number from the near design value of 6.8 noted previously (Figure 1). A highly non-uniform flow was calculated for these conditions, as shown in Figure 2a for the Mach number contours. The non-dimensionalized total temperature contours for this solution indicate a severe loss of energy from the gas into the wall. Total temperatures of the order of 60% of the stagnation value are seen near the centerline of the exit of the nozzle.

Several parameters changed for the solution shown in Figure 2. Gamma is now a field variable rather than a fixed value and contours are shown in Figure 2c. A variation from approximately 1.28 in the settling chamber at the stagnation condition to a value near 1.36 near the outflow is observed. It is not clear from the solutions shown in Figures 1 and 2 that the difference between the loss in total temperature is associated with changing from the ideal gas to the real gas code or whether it is in fact related somehow to the variation in the geometry calculated for these two cases.

Another hypothetical nozzle was calculated using the high enthalpy conditions

and a simple conical transition from the large throat to the test section radius. Results of these calculations are shown in Figures 3 and 4. Figure 3 presents the solution for the ideal gas case, with a gamma frozen at 1.28. Again, Mach number contours indicate an exit value that is continuously varying from the wall to near the centerline, where the maximum Mach number is approximately 4.6. The total temperature contours for this case are shown in Figure 3b, and again, indicate that the highest stagnation temperature is approximately 50% of that in the settling chamber. Thus it appears that the conditions shown in Figures 2 and 3 are consistent. Similar high energy losses were found for the equilibrium real gas version of the code as shown in Figure 4. Again, a highly non-uniform exit flow is predicted, with exit Mach numbers varying from zero near the wall to approximately 5.2. Total temperatures are only approximately half of those existing in the stagnation region. Very few differences are calculated in the flow fields for these simulations using either the ideal gas with frozen flow or equilibrium real gas codes. Values of gamma (Figure 4c) for the conical nozzle using the equilibrium real gas code vary from near 1.28 in the stagnation region to approximately 1.37 near the outflow.

In summary, it appears that both the contoured and the conical nozzles with large throats produce highly non-uniform exit flow fields because these nozzles are not designed to be wave cancelling to produce a uniform outflow. In addition, large energy losses are shown for all the cases with the large throat and high enthalpy conditions, indicating that the turbulent boundary layer might engulf the entire nozzle flow.

In order to investigate the potential source of these non-uniformities, a case was run with the ideal gas version for a laminar boundary layer. These results are shown in Figure 5 and indicate a relatively thick (but well defined) laminar boundary layer, but a core flow with expansion and compression waves leading to a very poor quality outflow condition. These waves are surely due to the non-cancelling walls of both the large-throat contoured nozzle (Figure 2) and the conical nozzle. The effect of these non-uniformities on the turbulent boundary layer development in the simulations is non-negligible and leads to unexpectedly thick layers with large energy losses.

The conical nozzle was recalculated using the ideal gas code with a Cebeci-Smith turbulence model in place of the Baldwin-Lomax model and the results are shown in Figure 6. This solution is quite different from that shown in Figure 3. A relatively thin turbulent layer is present up to a streamwise location that corresponds to the beginning of the highly non-uniform core flow shown in Figure 5 for the laminar case. Downstream of that location, the Cebeci-Smith boundary layer grows rapidly, decreasing the Mach number and losing energy to the cold wall.

Thus, it appears that the abnormally thick turbulent layers shown in previous solutions are a product of the implementation of the turbulence models and the non-uniform (vortical) core flows. In the numerical simulations, the eddy viscosity is applied to all vorticity to produce shear stresses, leading to an ever thickening boundary layer. In the physical flow, the vortical inviscid flow external to the boundary layer will be unstable (particularly when the local pressure gradient relaxes from its large negative value at the throat) and might be rapidly engulfed into a

9

growing boundary layer. This latter speculation might account for the very thick (up to 50% of the outflow radius) turbulent boundary layers observed in conical nozzle facilities.

IV. CONCLUSIONS

An investigation using computational fluid dynamics (CFD) has been carried out to examine the expected flow fields that will occur within the NASA-Ames reflected-shock wind tunnel nozzle. A simulation of the originally designed nozzle indicates a plausible flow field, while simulations involving a much larger throat producing a lower exit Mach number all resulted in highly distorted flow fields from either using a set of contours with a large throat faired into the originally designed wave cancellation section or from a newly conceived conical flow nozzle. Results were similar for the ideal gas code and the equilibrium real gas code. A solution using laminar flow was obtained that indicated a very non-uniform flow for the large throat conical nozzle. Additional calculations using yet another turbulence model indicated that the non-cancelling walls of these large throat nozzles and attendant non-uniformities might lead to rapid boundary layer growth in the simulations.

Although the simulations presented here are all accompanied by the uncertainty of the effects of the turbulence model, the qualitative features of the flow fields must be assumed to be correct in that they point to problematic areas associated with attempting to adapt the existing upstream and downstream hardware with arbitrarily conceived intermediate nozzles, and caution should be exercised when carrying out such designs for modifications to the existing facility.

Mach 7 Shock Tube Nozzle, Beta=1.02 mesh s17a Btime=0.200
Pt=215173psf Tt=6102R Tw=1000R Gamma=1.244 130000 iters

M_∞	0.100
α	0.00°
Re	1.41×10^7
Time	1.06×10^{-2}
GRID	201×31

CONTOUR LEVELS
0.000000
0.200000
0.400000
0.600000
0.800000
1.000000
1.200000
1.400000
1.600000
1.800000
2.000000
2.200000
2.400000
2.600000
2.800000
3.000000
3.200000
3.400000
3.600000
3.800000
4.000000
4.200000
4.400000
4.600000
4.800000
5.000000
5.200000
5.400000
5.600000
5.800000
6.000000
6.200000
6.400000
6.600000
6.800000
7.000000
7.200000
7.400000
7.600000

FIGURE 1 Flow field calculations for the original Mach 7 nozzle at the moderate enthalpy conditions: $Tt = 6102°R$, $Pt = 1500$ psi, $T_{wall} = 1000°R$ for ideal gas with gamma frozen at 1.244.

a) Mach number contours for full nozzle shown to correct scale.

MACH NUMBER

Mach 7 Shock Tube Nozzle, Beta=1.02 mesh s17a Btime=0.200
Pt=215173psf Tt=6102R Tw=1000R Gamma=1.244 130000 iters

M_∞ 0.100
α 0.00°
Re 1.41×10^7
Time 1.06×10^{-2}
GRID 201×31

CONTOUR LEVELS
0.00000
0.20000
0.40000
0.60000
0.80000
1.00000
1.20000
1.40000
1.60000
1.80000
2.00000
2.20000
2.40000
2.60000
2.80000
3.00000
3.20000
3.40000
3.60000
3.80000
4.00000
4.20000
4.40000
4.60000
4.80000
5.00000
5.20000
5.40000
5.60000
5.80000
6.00000
6.20000
6.40000
6.60000
6.80000
7.00000
7.20000
7.40000
7.60000

FIGURE 1   (Continued)

    b)   **Mach number contours for nozzle shown with expanded radial coordinate.**

NORMALIZED STAGNATION TEMPERATURE
Mach 7 Shock Tube Nozzle, Beta=1.02 mesh s17a Btime=0.200
Pt=215173psf Tt=6102R Tw=1000R Gamma=1.244 130000 iters

M_∞ 0.100
α 0.00°
Re 1.41×10^7
Time 1.06×10^{-2}
GRID 201×31

CONTOUR LEVELS
0.15000
0.17500
0.20000
0.22500
0.25000
0.27500
0.30000
0.32500
0.35000
0.37500
0.40000
0.42500
0.45000
0.47500
0.50000
0.52500
0.55000
0.57500
0.60000
0.62500
0.65000
0.67500
0.70000
0.72500
0.75000
0.77500
0.80000
0.82500
0.85000
0.87500
0.90000
0.92500
0.95000
0.97500
1.00000
1.02500

x

y

FIGURE 1   (Concluded)

c)   Normalized stagnation temperature contours.

FIGURE 2 Flow field calculations for the large throat contoured nozzle at the high enthalpy conditions: $T_t = 9460^\circ R$, $P_t = 8125$ psi, $T_{wall} = 600^\circ R$ for equilibrium real gas code.

a) Mach number contours.

TOT. TEMPERATURE Shock Tube Nozzle v2 Equilibrium Real Gas
It=9460R Pt=117000psf Tw=600R Bt=.55 264000it Nshift=150

1400
1800
2200
2600
3000
3200

1800
2600
3200
3800

x

y

-1.0 0.0 1.0 2.0 3.0 4.0 5.0 6.0 7.0 8.0 9.0

0.8 0.6 0.4 0.2 0.0 -0.2 -0.4

CONTOUR LEVELS
200.000
400.000
600.000
800.000
1000.00
1200.00
1400.00
1600.00
1800.00
2000.00
2200.00
2400.00
2600.00
2800.00
3000.00
3200.00
3400.00
3600.00
3800.00
4000.00
4200.00
4400.00
4600.00
4800.00
5000.00
5200.00
5400.00

FIGURE 2   (Continued)

b)   **Total temperature contours in degrees Kelvin (°K).**

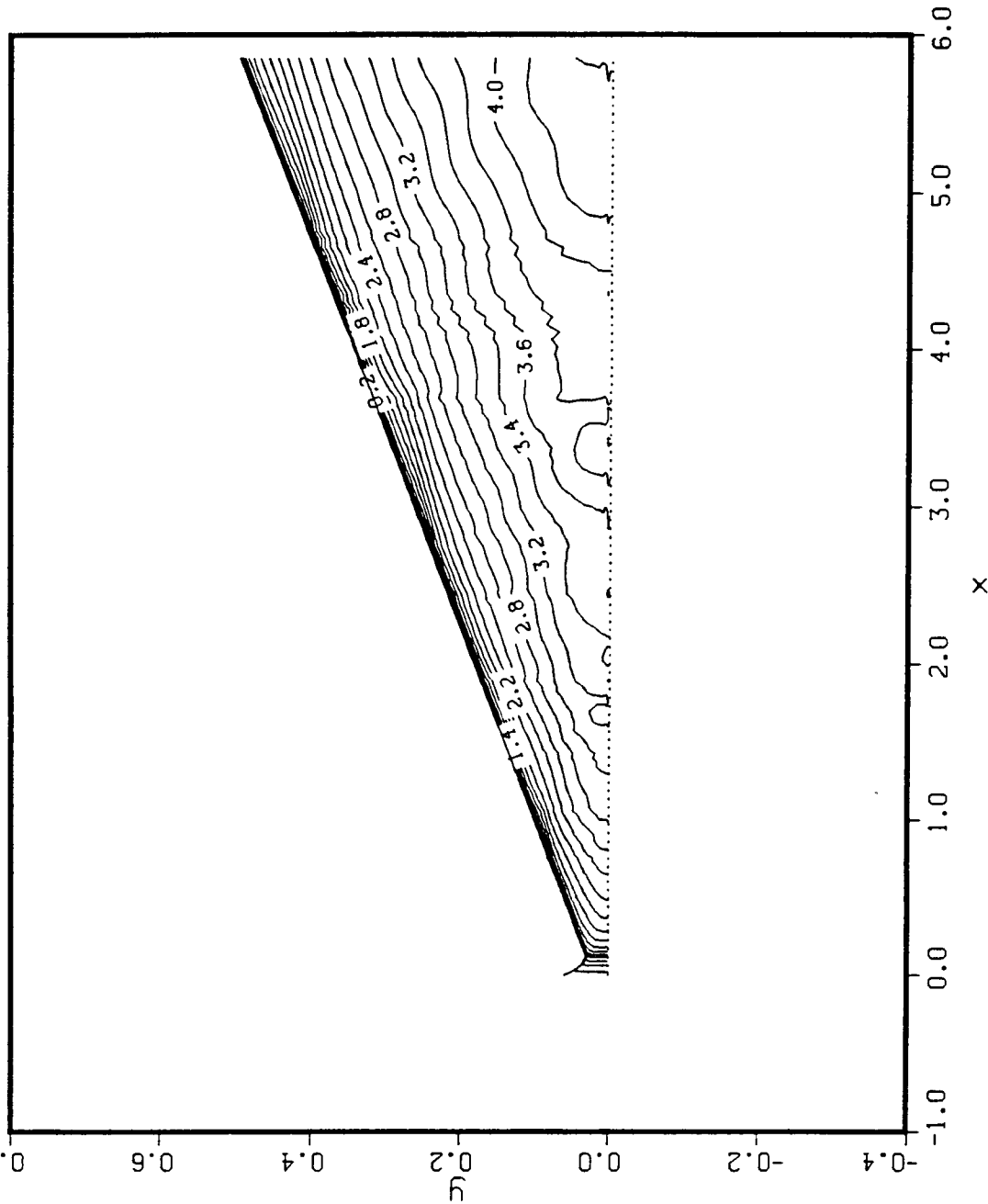FIGURE 2 (Concluded)

c) Isentropic exponent (gamma) contours.

FIGURE 3 Flow field calculations for the conical nozzle at the high enthalpy conditions: $T_t = 9460°R$, $P_t = 8125$ psi, $T_{wall} = 600°R$ for ideal gas with gamma frozen at 1.28.

    a)   **Mach number contours.**

NORMALIZED STAGNATION TEMPERATURE

Shock Tube Nozzle V.3, Ideal Gas s12a Btime=0.502
Tt=9960R Pt=117000psf Tw=600R Gamma = 1.28 140000it

M_∞ 0.155
α 0.00°
Re 7.66×10^7
Time 1.36×10^{-2}
GRID   201×31

CONTOUR LEVELS
0.05000
0.10000
0.15000
0.20000
0.25000
0.30000
0.35000
0.40000
0.45000
0.50000
0.55000
0.60000
0.65000
0.70000
0.75000
0.80000
0.85000
0.90000
0.95000
1.00000
1.05000

FIGURE 3   (Concluded)   b)   Normalized stagnation temperature.

CONTOUR LEVELS
0.00000
0.20000
0.40000
0.60000
0.80000
1.00000
1.20000
1.40000
1.60000
1.80000
2.00000
2.20000
2.40000
2.60000
2.80000
3.00000
3.20000
3.40000
3.60000
3.80000
4.00000
4.20000
4.40000
4.60000

MACH. No.   Shock Tube Nozzle v3 Equilibrium Real Gas
Tt=9460R Pt=1170000psf Tw=600R Bt=.369 166000it Nshift=100

M_∞ 0.146
α 0.00°
Re   7.22×10⁷
Time   1.06×10⁻²
GRID   201×31

y

x

-1.0   0.0   1.0   2.0   3.0   4.0   5.0   6.0

0.8   0.6   0.4   0.2   0.0   -0.2   -0.4

1.4 2.2 2.8 3.2 3.4 3.6 0.2 1.8 2.4 2.8 3.2 4.0

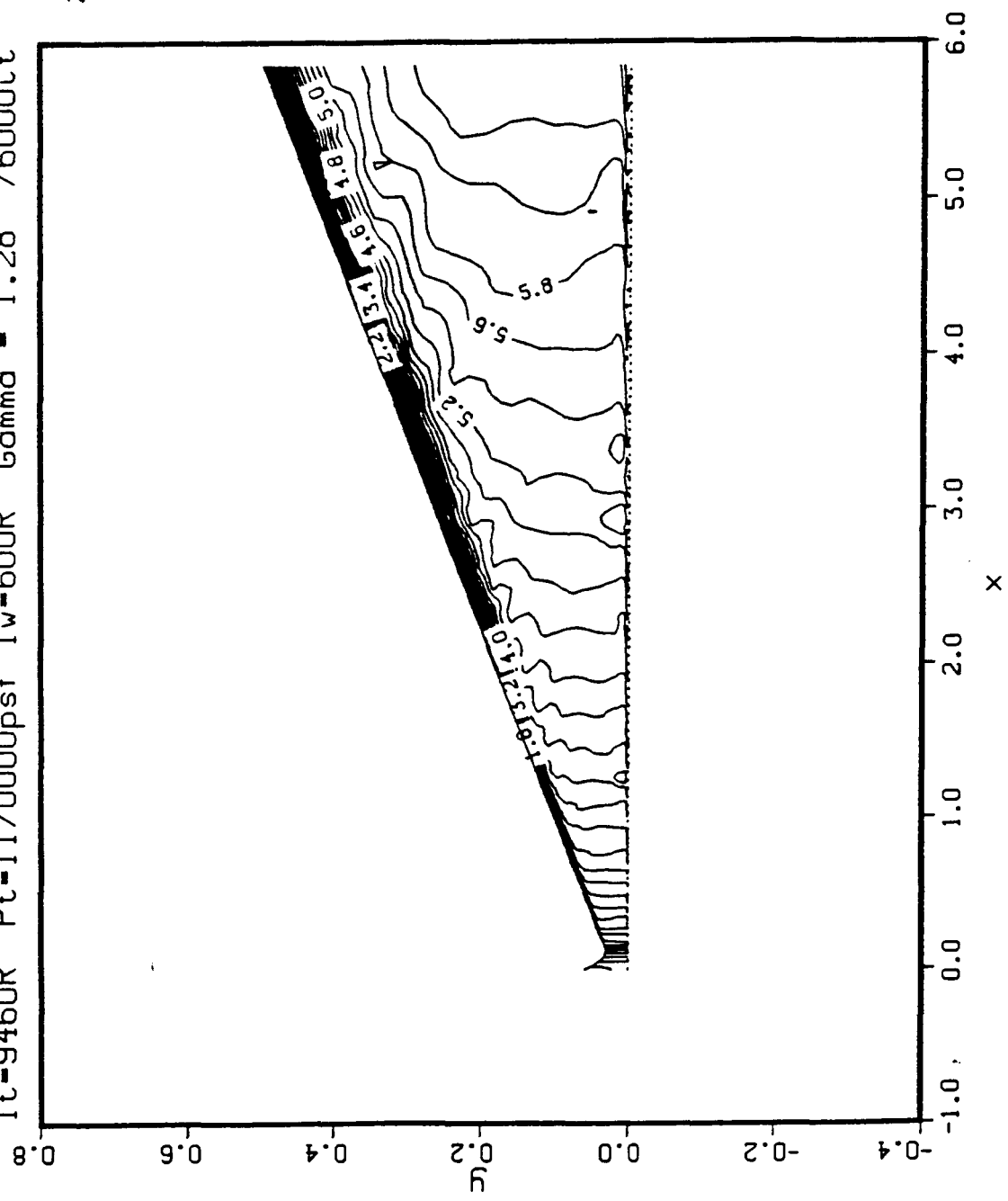**FIGURE 4**   **Flow field calculations for the conical nozzle at high enthalpy conditions: T_t = 9460°R, P_t = 8125 psi, T_{wall} = 600°R using the equilibrium real gas code.**

a)   Mach number contours.

TOT. TEMPERATURE   Shock Tube Nozzle v3 Equilibrium Real Gas0.146   M∞

It=9460R Pt=117000psf Tw=600R Bt=.369 166000it Nshift=100   0.00°   α

Re   7.22×10⁷

Time   1.06×10⁻²

GRID   201×31

CONTOUR LEVELS
200.000
400.000
600.000
800.000
1000.00
1200.00
1400.00
1600.00
1800.00
2000.00
2200.00
2400.00
2600.00
2800.00
3000.00
3200.00
3400.00
3600.00
3800.00
4000.00
4200.00
4400.00
4600.00
4800.00
5000.00
5200.00
5400.00

FIGURE 4   (Continued)

b)   Total temperature contours in °K.

GAMMA   Shock Tube Nozzle v3 Equilibrium Real Gas

M_∞ 0.146
α 0.00°
Re 7.22×10^7
Time 1.06×10^{-2}
GRID   201×31

Tt=9460R Pt=117000psf Tw=600R Bt=.369 166000it Nshift=100

x

y

CONTOUR LEVELS
1.28500
1.29000
1.29500
1.30000
1.30500
1.31000
1.31500
1.32000
1.32500
1.33000
1.33500
1.34000
1.34500
1.35000
1.35500
1.36000
1.36500
1.37000
1.37500
1.38000
1.38500
1.39000
1.39500
1.40000

FIGURE 4   (Concluded)
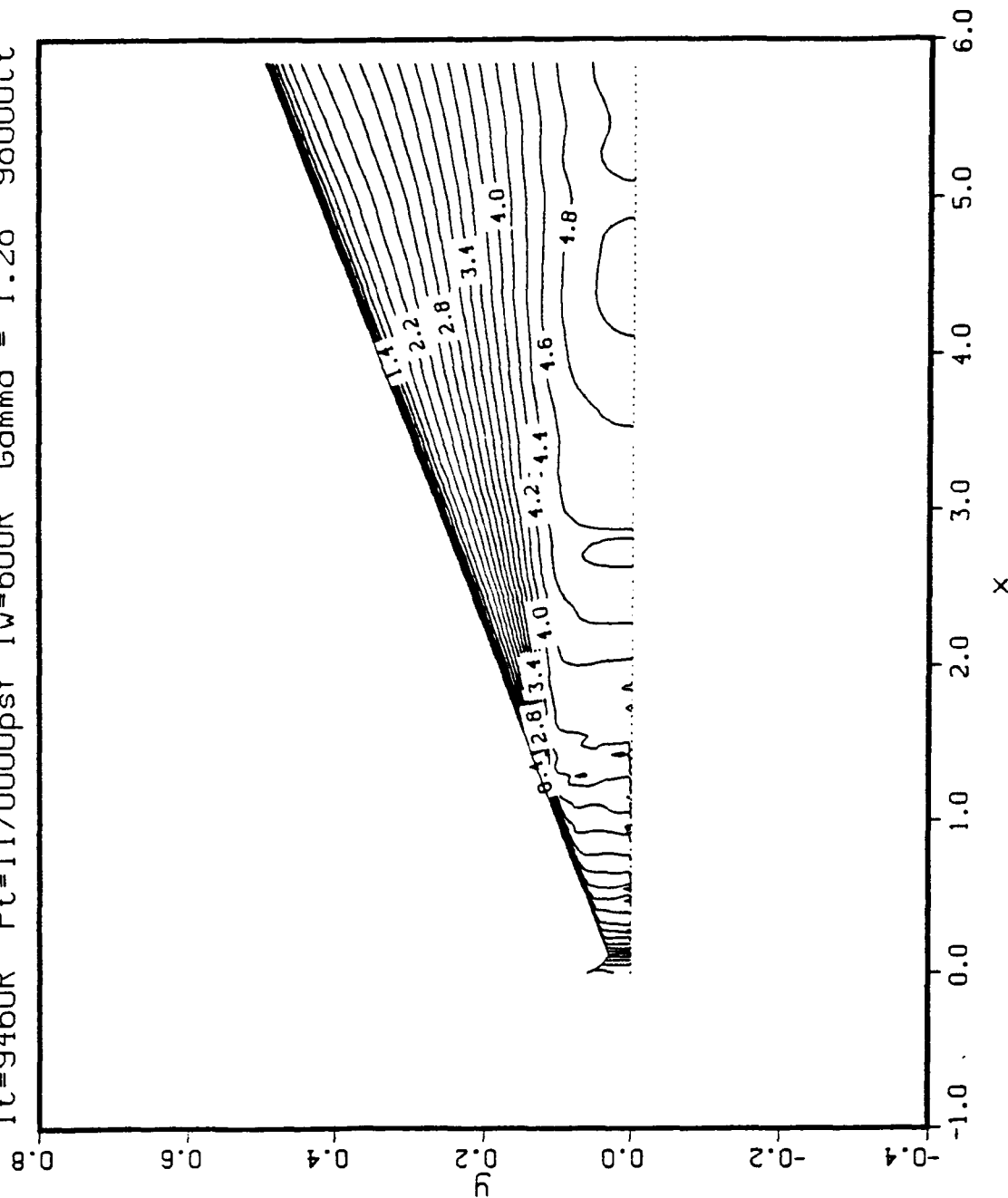
c)   **Isentropic exponent (gamma) contours.**

FIGURE 5 Flow field calculations for the conical nozzle with high enthalpy conditions: $T_t = 9460^{\circ}R$, $P_t = 8125$ psi, $T_{wall} = 600^{\circ}R$ for ideal gas with gamma frozen at 1.28 for laminar flow.

    a)   **Mach number contours.**

NORMALIZED STAGNATION TEMPERATURE
Shock Tube Nozzle V.3, Ideal Gas Laminar s8a Bt=.19
Tt=9460R Pt=117000psf Tw=600R Gamma = 1.28 76000it

M∞   0.194
α   0.00°
Re   9.50×10^7
Time 4.05×10^{-3}
GRID 201×31

CONTOUR LEVELS
0.05000
0.10000
0.15000
0.20000
0.25000
0.30000
0.35000
0.40000
0.45000
0.50000
0.55000
0.60000
0.65000
0.70000
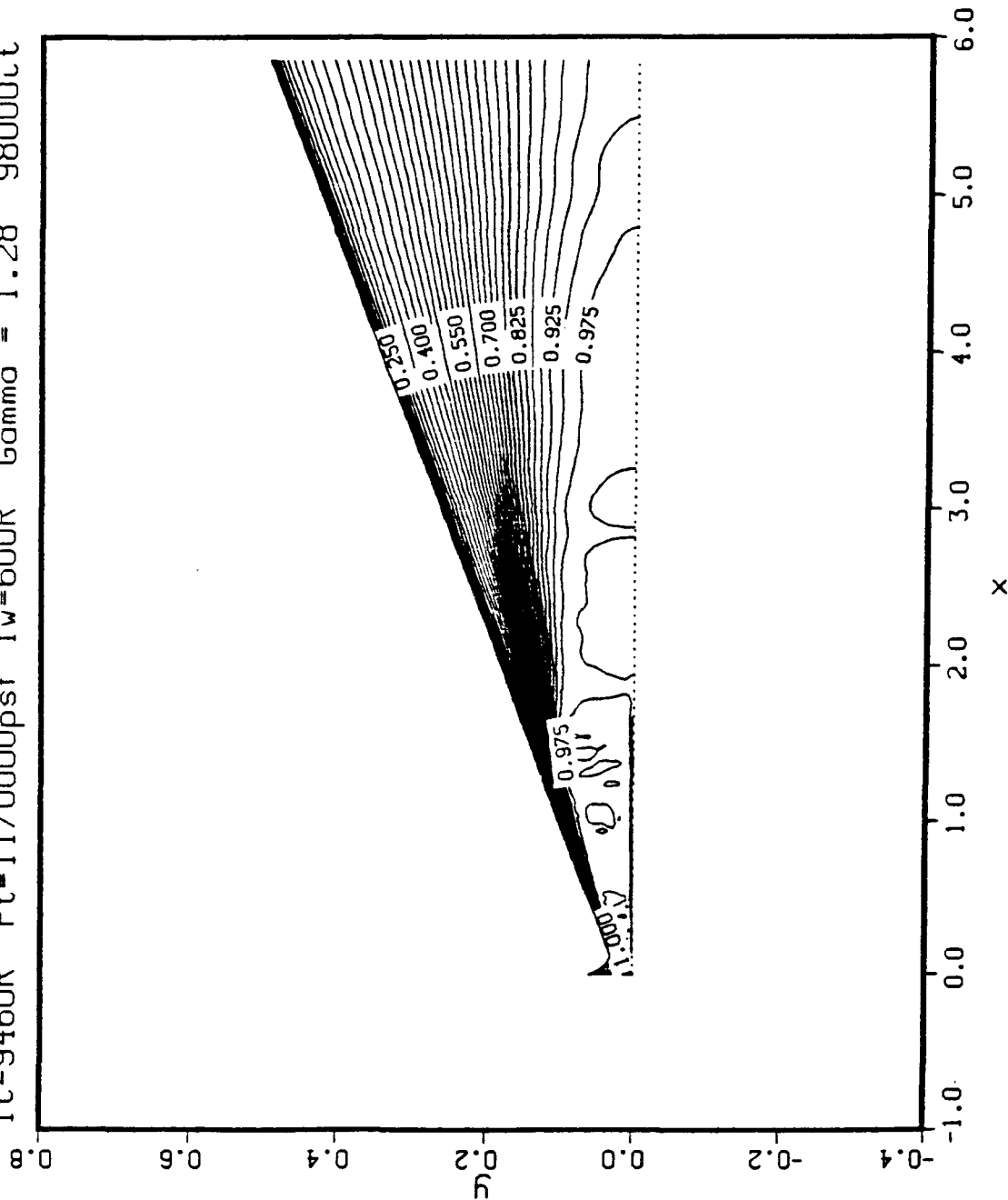0.75000
0.80000
0.85000
0.90000
0.95000
1.00000
1.05000

FIGURE 5   (Concluded)

b)   Normalized stagnation temperature contours.

FIGURE 6 Flow field calculations for the conical nozzle with high enthalpy conditions: $T_t = 9460°R$, $P_t = 8125$ psi, $T_{wall} = 600°R$ for ideal gas with gamma frozen at 1.28 with Cebeci-Smith eddy viscosity model.

a) Mach number contours.

FIGURE 6 (Concluded) **b) Normalized stagnation temperature contours.**